Glenn Research Center

Introduction to Wind-US

Dr. John Slater
John.W.Slater@nasa.gov
(216) 433-8513

Dr. Charlie Towne
Charles.E.Towne@nasa.gov
(216) 433-5851

NASA Glenn Research Center
Brook Park, Ohio

TFAWS 2007 - September 10, 2007

Glenn Research Center

Objectives

- Provide some background on the models, algorithms, and methods used in the Wind-US CFD code for analyzing internal and external compressible flows.
- Provide basic instruction on the use of the Wind-US CFD code as a tool for performing flow field analysis. This is a “Getting Started” class.
- “Hands On” session to try out Wind-US.

Glenn Research Center

Scope

- CFD from perspective of capabilities of the Wind-US code
- CFD for aerodynamic analysis (p, T, τ)
- Perfect gas (air) $\rightarrow$ chemically reacting multi-species gas
- Compressible flow (0.1 < Mach < 10+)
- Steady and unsteady (time-varying) flow
- Inviscid, laminar, or turbulent flow
- External and internal flow
- Special models for propulsion simulation (bleed, VGs, etc…)
- Multi-zone, Structured or Unstructured Grid

Glenn Research Center

Outline

- Introduction
- Examples of CFD Applications using Wind-US
- NPARC Alliance and the NPARC Flow Simulation System
- CFD Analysis Process
- Flow Field Problem Formulation
- Geometry Modeling, Flow Domain Modeling, and Grid Generation
- Physical and Zonal Boundary Conditions
- Setup and Execution of the CFD Simulation
- Conducting and Reporting the Results of a CFD Simulation
- Tutorial Case Demonstration
- Wind-US Demonstration and Hands-On Examples